EAN 2019
57th conference on experimental stress analysis

The validation of numerical models of potted inserts in sandwich structures based on experimental testing

CABRNOCH Bohuslav[1,a], KRÁL Michal[1,b] and KRUML Josef[1,c]

[1]Czech Aerospace Research Centre, Department of Composite Technologies, Beranových 130,199 05 Praha - Letňany, Czech Republic

[a]cabrnoch@vzlu.cz, [b]kral@vzlu.cz, [c]kruml@vzlu.cz

Keywords: spacecraft, potted insert, finite element method, experimental testing, stiffness

Abstract. The main goal of this paper is to describe the research of an appropriate approach to modelling of potted inserts in sandwich panels used in spacecraft structures. Potted inserts are used for the connection of sandwich panel with other parts of the assembly. In other words, they are used for the introduction of forces into sandwich panel. The main goal of the research was not to find the most accurate manner of modelling but to find an optimal manner within the meaning of compromise between accuracy of prediction and model complexity. The main observed property was the out-of-plane stiffness of the insert. This property affects global behaviour of the structures besides in terms of natural frequencies which is one of the key observed characteristics of spacecraft structures.

Introduction

Sandwich structures are frequently used in many industrial sectors especially in the design of rolling stocks and spacecraft structures. Sandwich structures consist of two thin skins with high in-plane strength and stiffness connected by the lightweight core with high out-of-plane shear stiffness. This combination ensures excellent stiffness-to-weight proportion in bending. To utilize this benefit, it is necessary to ensure a sufficient design solution of introduction of forces into the sandwich panel. This problem is very often solved by potted inserts. Based on requirements of individual structure, the insert is designed as partially potted, fully potted or as through the whole thickness insert. During FE modelling of sandwich structures, it must be ensured that model will predict the stiffness and strength of potted insert with sufficient precision. The main goal of this paper is to describe experimental validation of created FEM models, which are used for modelling of panels used in spacecraft structures. These structures are subjected to static and dynamic environments, where the stiffness of the potted inserts significantly influences the global behaviour of the structure.

The main goal of the presented paper is to find optimal proportion between the size of FE model in terms of number of elements and accuracy of predicted potted insert stiffness in out-of-plane direction. The requirements to potted insert model are different for individual analysis types. Linear static analysis is usually performed to determine stress-strain response of the structure and thanks to high computing capacity it is possible to model the structure in detail. Other analysis types like modal analysis or random response are more demanding to computational capacity and the size of FE model should stay on a satisfactory level.

Review of current state of the art

Design and sizing principles of sandwich structures with potted inserts in terms of structural strength is usually based on methodologies described in design handbooks, see e.g. [1]. These handbooks provide sufficient sizing methods. Numerical modelling of damage and growth of individual damage mechanisms is very often the objective of research projects and can provide a good tool for the understanding of damage mechanisms evolution. The research of damage evolution using different types of FE models is performed e.g. in [2], where the main objective is to compare different approaches to discretization of honeycomb core. Core is usually modelled as 3D continuum using volume elements. Authors have shown that better estimation of response of potted inserts is achieved by modelling honeycomb in detail using 2D elements. Undoubtedly, detailed honeycomb model is better for correct modelling of damage growth. Anyway, in engineering practice, detailed modelling seems to be highly time consuming and it is necessary to find a compromise between precision and time requirements. The accuracy can be satisfactory also using 3D continuum model of the core as shown in [3].

FEM models

To evaluate the effect of the number of elements, more FE models were built and further predicted stiffness was compared with experimental data. Models were created in Femap pre-processor. The example of created FE model is presented in Fig. 1. Two planes of symmetry were taken into account. The distribution of load was performed using RBE3 element. The core was modelled using 3D elements using 3D orthotropic material model. Skins were modelled using laminate type element as 2D orthotropic material model. Insert and resin were modelled as 3D isotropic continuum using 3D elements. Analysis was performed as a linear static using Nastran solver.

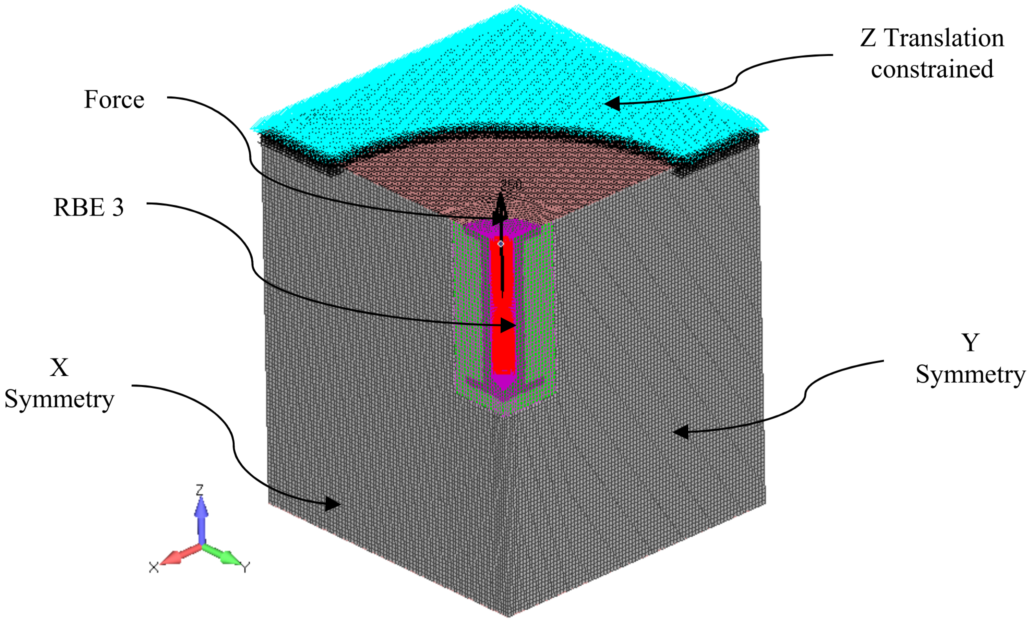

Fig. 1: Example of created FE model with two plane of symmetry

FE model presented in Fig. 1 is highly complex with 675 209 3D volume elements and 13 586 2D laminate elements. Other model with lower complexity was created. In this case 772 3D volume elements and 378 2D laminate elements were used. Both models are compared in Fig. 2.

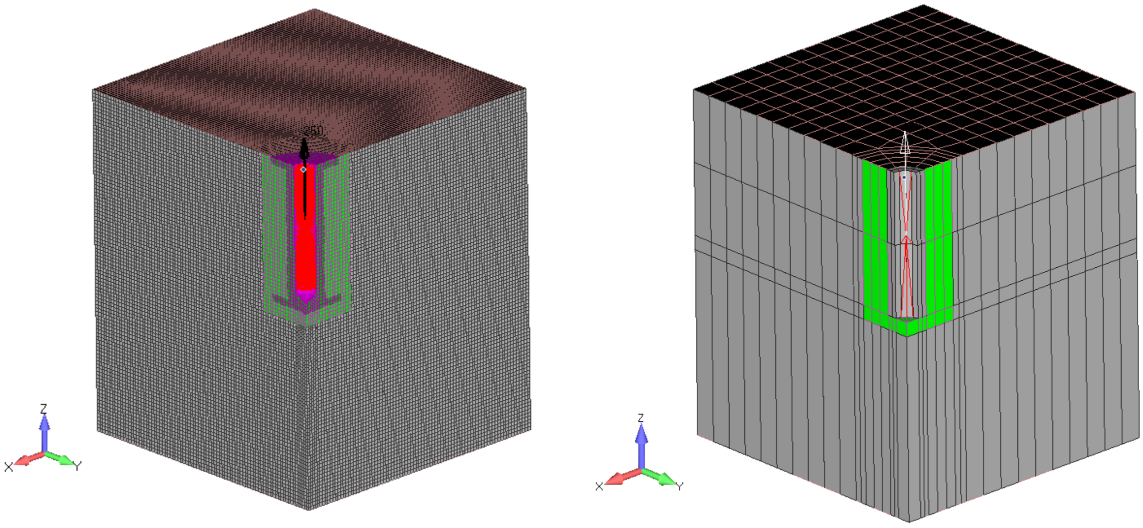

Fig. 2: Detailed FE model of potted insert (left one) and model with the lower number of elements (right one)

Experimental arrangement

The setup of experimental testing is presented in Fig. 3. Testing was performed using Instron test machine. Load rate was 2 mm/min. Loading was stopped when critical load has been reached, i.e. at 2 mm deflection of the skin.

Fig. 3: Detailed FE model of potted insert (left one) and model with the lower number of elements (right one)

Displacements were measured using optical system Q400 DIC - Digital Image Correlation System by Dantec Dynamics, see [4]. Digital image correlation is a full field image analysis method based on grey digital images. This experimental method allows determining displacements of observed object in three dimensions. Speckle pattern was made by spraying. See Fig. 4, where the boxes with recognizable pattern are indicated. Displacements were measured both on test specimen and on the jig to obtain results without affecting by the softness of the jig.

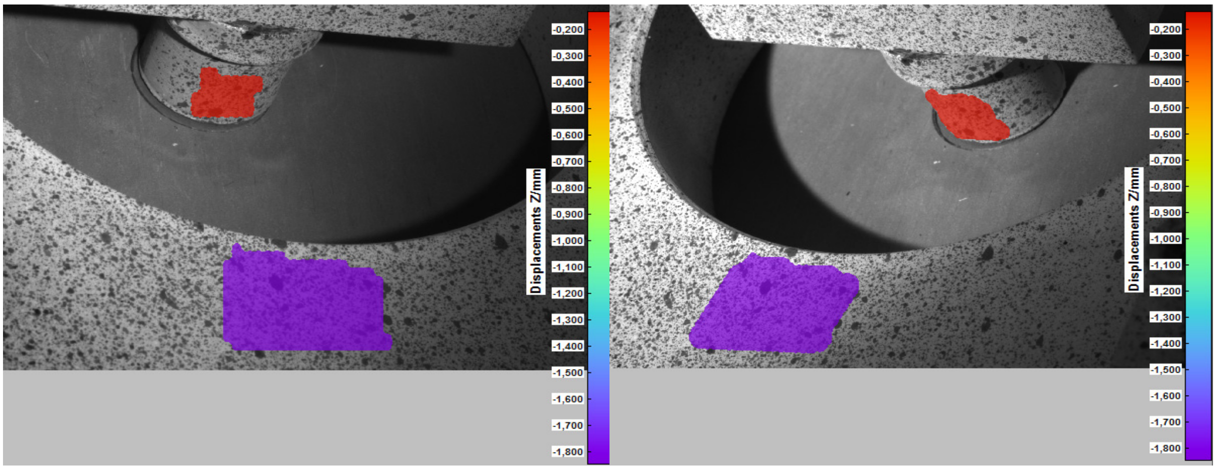

Fig. 4: Designation of boxes with recognizable pattern on test specimen and on the jig

Results

Measured characteristic of insert stiffness are presented as a dependence of displacement and force, see Fig. 5. Stiffness was evaluated as secant stiffness on the force base 0.5 kN – 1.5 kN. Five test specimens were tested for each design solution of sandwich or insert. Example of test specimens is shown in Fig. 6. Evaluated stiffness of inserts with statistical characteristics is shown in Tab. 1. Mean value is 26602 N/mm. It is obvious that the results show a great dispersion. The value of correlation coefficient is 37.5%. This value is caused also by a small statistical sample.

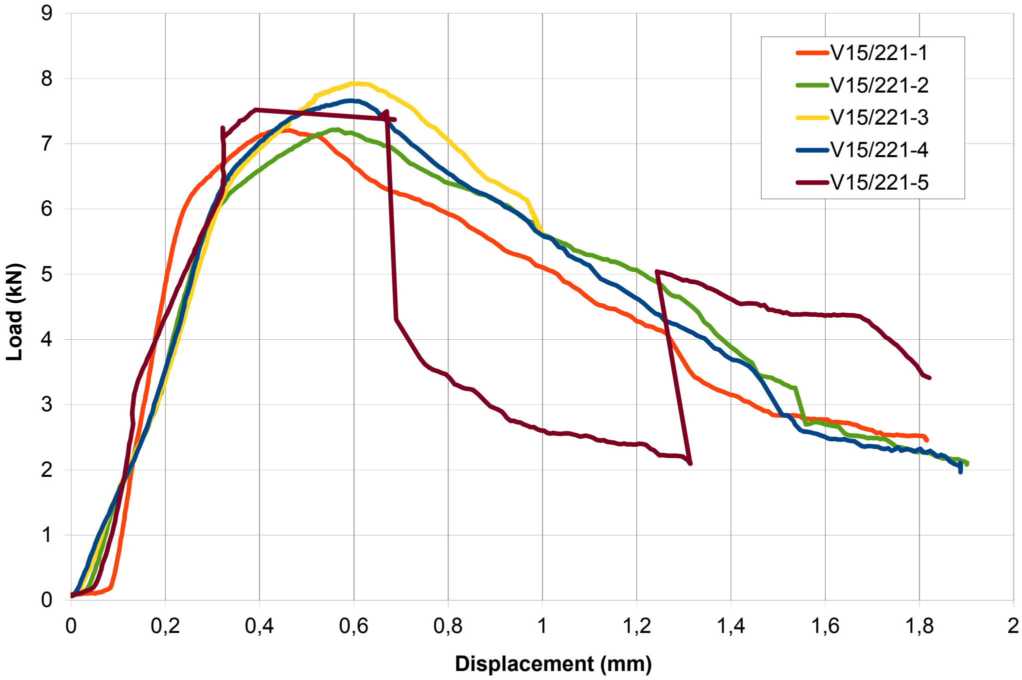

Fig. 5: Results of testing of five specimens V15/221-1 - V15/221-5

Generally it is recommended to use more test specimens. But in order to perform a basic validation of FE model, five specimens is a satisfactory quantity.

Stiffness predicted by FE models is shown in Tab. 2. It is obvious that the complexity of FE model does not have a significant effect on out-of-plane stiffness prediction. This was an important finding because it was proofed that the complexity of FE model can be changed if needed without affecting of results. Also the conformity between experimental data and numerical prediction is satisfactory. The fraction of experiment and simulation is 0.82 for the model with high complexity and 0.79 for model with lower complexity. Numerical prediction can be improved by defining of contact problem between test specimen and jig. Used boundary conditions, defined in Fig. 1, do not allow the creation of the gap between test specimen and jig which is not in the line with reality and causes higher stiffness of the model.

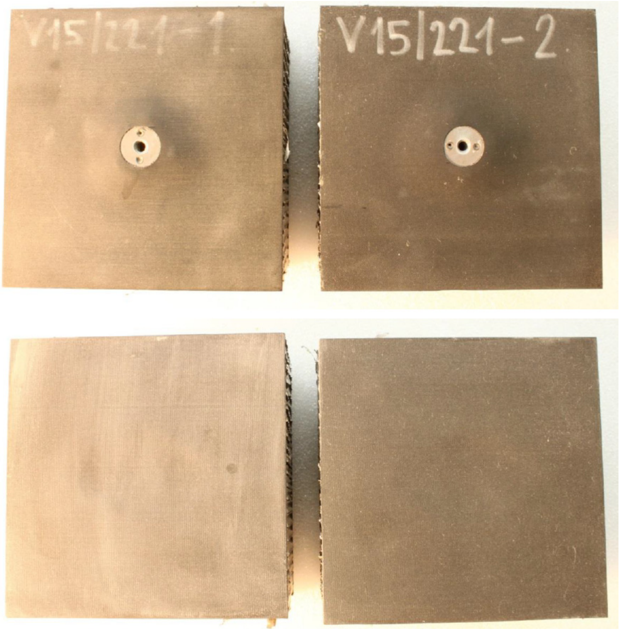

Fig. 6: Two examples of used test specimens after the test - bottom and upper side is shown

Tab. 1: Experimental results

Test specimen	Stiffness [N/mm] 0.5 - 1.5 kN
V15/221 - 1	42853
V15/221 - 2	24639
V15/221 - 3	20067
V15/221 - 4	17451
V15/221 - 5	28000
Mean [N/mm]	26602
St. Deviation [N/mm]	9954,3
Variation Coef. [%]	37,5

Tab. 2: Numerical results

Numerical simulation	Predicted Stiffness [N/mm]
High complexity model	32479
Low complexity model	33412

Conclusions

It is obvious that experimental testing still plays a very important role in the development of aerospace and other structures and in validation of numerical models. The development of spacecraft structures is always associated with detailed testing and verification. The reasons are as follows - high cost of spacecraft structures, very low number of structures already built and therefore resulting low practical experience.

The attention was concentrated to out-of-plane stiffness of inserts. In next step also in-plane stiffness will be verified because in-plane stiffness has also important effect on the behaviour of the sandwich structure. It can be stated that the effect of the number of elements on predicted stiffness is not as significant as expected and the difference is up to 5% which is acceptable.

Acknowledgements

This research was performed with financial institutional support from government budget through the Ministry of Industrial and Trade of the Czech Republic.

References

[1] ESA European Space Agency. Space Engineering – Insert Design Handbook. ESA Requirements and Standards Division, March 2011. ECSS-HB-32-22A.

[2] R. Seeman, D. Krause, Numerical modelling of partially potted inserts in honeycomb sandwich panels under pull-out loading, Composite Structures. 2018 (203) 101-109.

[3] P. Bunyawanichakul, B. Castanié, J. J. Barrau, Non-linear finite element analysis of inserts in composite sandwich structures, Composites Part B: Engineering. 2008, 39(7-8) 1077-1092.

[4] Dantec Dynamics, Q-400 µDIC^{TM} Microscopic Shape, Displacement & Strain Measurement, Publication no. 354_v1, 2015, Dantec DynamicsGmbH.